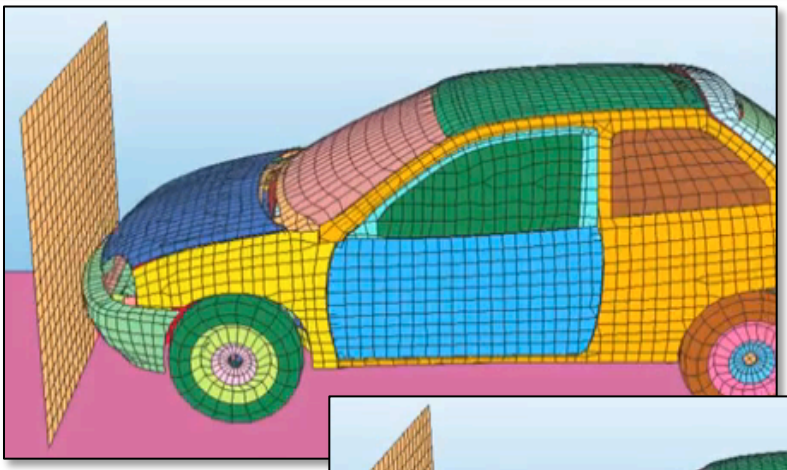
CEE570/CSE551 – Finite Element Methods
(In solid and structural mechanics)

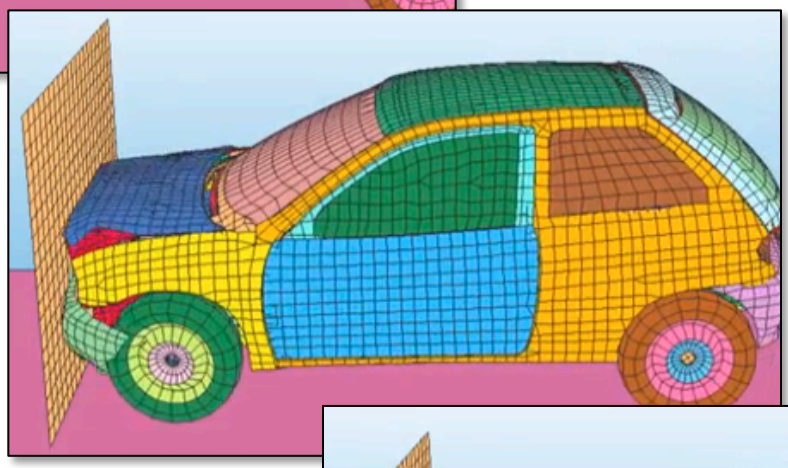
Spring 2014

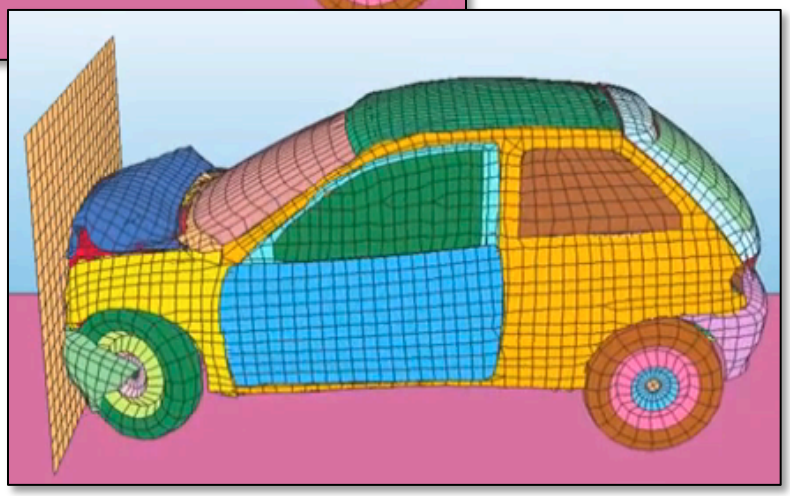
Patran and Abaqus Lab Notes



Car crash analysis with LS-DYNA

Livermore Software Technology Corporation
www.lstc.com





Watch the simulation on You Tube:

LS-DYNA: Car Crash

Instructor: Glaucio H. Paulino
Teaching Assistants: Heng Chi, Junho Chun, Sofie Leon, Tomas Zegard, Shelly Zhang

Patran 2010

User's Guide

MSC Software

2 Fundamentals

- Starting and Exiting Patran 32
- A Tour of the Patran Interface 33
- How to Get Things Done 34

Starting and Exiting Patran

Patran (Windows):

- Start up Patran by choosing

  Start/Programs/MSC.Software/Patran 200x.

  or

  Double-click on the Patran 200x icon on the desktop if installed using this option.

We will use Unix on the EWS machines

Patran (UNIX):

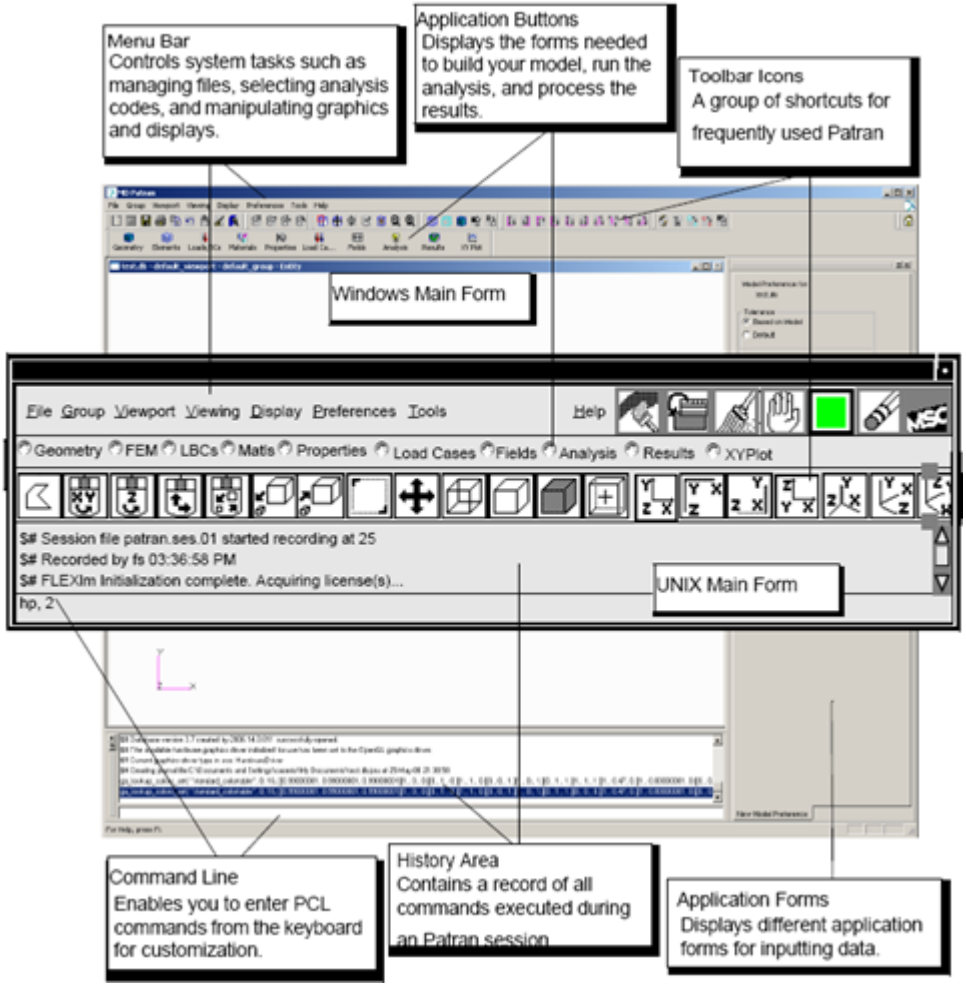
- Type `patran` and press <Return> at the operating prompt from any UNIX shell window. This assumes that the command `patran` is in the user's path.

If you do not see the Patran window or if Patran does not start correctly after entering the start-up command, report the error to your system administrator or access your Patran *Installation and Operations Guide*. (You can access the *Installation and Operations Guide* on the Web at www.mscsoftware.com, under Support, Training, and Documentation.)
The *Installation and Operations Guide* contains step-by-step directions for tracking down start-up problems and provides a working solution.

To quit, from the Patran File menu, select Quit. Patran automatically saves any changes that you made to the current database when you exit.

A Tour of the Patran Interface

At the center of the Patran window is a blank graphics viewport where you construct your finite element analysis model. The menu bar, toolbar, and application buttons shown below are your control panel. The history area keeps you informed on what Patran is doing, and reports on what has occurred. The command line is used to enter customized commands and the application forms are used to build each part of your model. Differences between Windows and UNIX are noted.


Menu Bar
Controls system tasks such as managing files, selecting analysis codes, and manipulating graphics and displays.
Application Buttons
Displays the forms needed to build your model, run the analysis, and process the results.
Toolbar Icons
A group of shortcuts for frequently used Patran
Windows Main Form
File Group Viewport Viewing Display Preferences Tools
Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by fs 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2
UNIX Main Form
Command Line
Enables you to enter PCL commands from the keyboard for customization.
History Area
Contains a record of all commands executed during an Patran session
Application Forms
Displays different application forms for inputting data.

How to Get Things Done

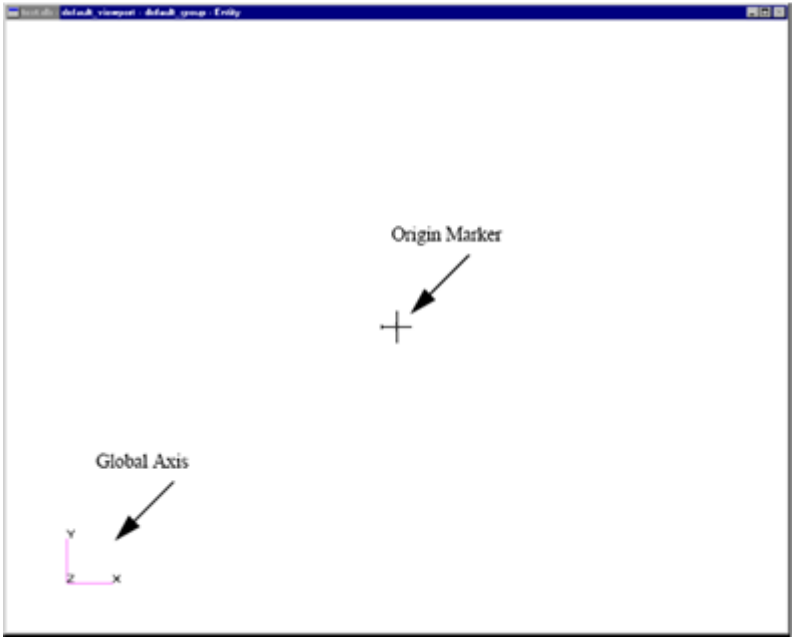
This section contains basic information on using the different components in the Patran to complete tasks.

- Working with viewports
- Menubar
- Application buttons and forms
- System and Toolbar icons
- Command and history areas
- Picking and selecting

This information is intended to get you started using Patran. More detailed information can be found in the Patran *Reference Manual*.

Working with Viewports

A viewport is a graphics window in which the entire model (or a selected portion of a model) appears. A blank viewport, similar to the one shown below, appears at the center of the Patran window when you create a new Patran database.



The top of the viewport lists the name of your database (in this case `test.db`), the name of the viewport (default_viewport), the name of the current group (default_group), and the mode of operation for

displaying the model (Entity). The Global Axes show the orientation of your model. A plus sign (+) marker indicates the global origin.

To move and resize viewports within the Patran window:

1. Move the cursor over the Viewport title bar.
2. Click and drag the viewport to a new location.
3. Window machines allow for double clicking to fit the graphics window.

or

1. Move the cursor to any of the outside boarders or corners of the viewport.
2. When a two-headed arrow appears, click and drag the viewport to the desired size.

With Patran, there is no limit to the number of viewports you can create. Multiple viewports enable you to see different views of the model, different parts of the model, and different analysis results.

To create multiple viewports:

1. Select Viewport/Create from the top menu bar.

   The Viewport Create application form appears in the Patran window.

2. Enter a name for the new viewport in the New Viewport Name textbox, and click Apply.

   Your original viewport is the default viewport, and each subsequent viewport that you create appears in the Existing Viewports listbox.

The Menu Bar

The Menu Bar is located directly under the Patran title bar. Each menu keyword has a drop-down menu with additional commands. The menu keywords are shown below.

File

When you invoke Patran, only the File menu is enabled; the other menus become enabled once you open a database. The File menu provides access to the various files used by Patran, including database file management, file importing and exporting, session file handling, hardcopy file creation, the status database, and session exiting.

Group

The Group menu contains various options for you to organize into groups the geometric entities (points, curves, surfaces, solids) and finite element entities (nodes, elements, multipoint constraints) of a Patran model. You can then perform numerous modeling and postprocessing tasks on these groups.

For example, you could group finite elements composed of different materials. Or, for viewing results, you could create separate groups for displaying temperature and stress distribution on the same model simultaneously.

Viewport

Viewports are movable, sizable graphics windows used to display all or part of a model. Each viewport provides one independent view of a stationary model, and maintains a set of parameters to determine how you want to view it.

Use the Viewport menu to create, edit, remove, and arrange your different viewports.

To move, resize, or create new viewports, see Working with Viewports.

Viewing

The Viewing menu contains various options to control the size and orientation of a model within a viewport. You can manipulate a model's rotational orientation, size, position, type of projection, scale factors, and clipping planes. You can also control your viewport's viewing plane, observer position, window center, and focal point.

Changing your view in no way alters the model. With the Viewing menu options, you can pan, zoom in and out of, rotate, and resize your model.

Viewing Size and Positions on the Toolbar

For quick changes in viewing orientation, select from the set of viewing icons on the toolbar. For additional information on the toolbar, see Viewing Functions.

Also note that as you construct your model, Patran automatically resizes the viewport each time you add an entity outside the current field of view. The viewport is adjusted to encompass all entities in your current group.

More About Viewing Rotation

There are two ways to view the rotation of your model: around the model's global axes or around your screen's axes. You can view each of these in absolute or relative terms, as shown in the table below:

Model Absolute	Rotation is about the global axes, starting from the axes' zero rotation point.
Model Relative	Rotation is about the global axes, starting from the axes' current location.
Screen Absolute	Rotation is about the screen axes, starting from the axes' zero rotation point.
Screen Relative	Rotation is about the screen axes, starting from the axes' current location.

Display

Databases can get large and full of geometric and finite element entities. The Display menu helps you organize and enhance the appearance of these entities. With the Display menu, you can determine which entities you want displayed and how you want them displayed. And you can control numerous visualization features, such as render style, entity plotting and erasing, highlighting, and label display.

The Display features do not affect the basic operations of Patran. They merely enhance the usability and appearance of your models.

Preferences

With the Preferences menu, you can choose what parameters you want to govern the construction and appearance of your model.

The Preferences menu defines global parameters that you can override in specific applications. For example, during an equivalence operation, you can override the global model tolerance (the default equivalencing tolerance). Patran, however, will not use the new tolerance value in subsequent operations outside the given application.

The only way to change a preference permanently is to set it within a Preferences form.

Tools

The Tools menu executes the optional applications licensed at your site without exiting Patran. It also provides a path to some of Patran's newer, add-on features that you purchased at an additional cost. If an application module is not licensed, the option for it on the Tools menu will be dimmed, indicating that it is unselectable.

Using Lists

You can create numerous types of lists using the List option, including the following:

- You can, for example, build a list of entities having all nodes that equal zero.
- You can also combine lists to create other lists; for example all nodes equal to a certain value, and all elements associated with that value.
- You can use List to perform Boolean operations.

Help

Use the Help menu to retrieve detailed online documentation for all of Patran's features and tools. Through the Help menu you can also access tips on keyboard shortcuts, mouse functions, and screen picking.

Application Buttons and Application Forms

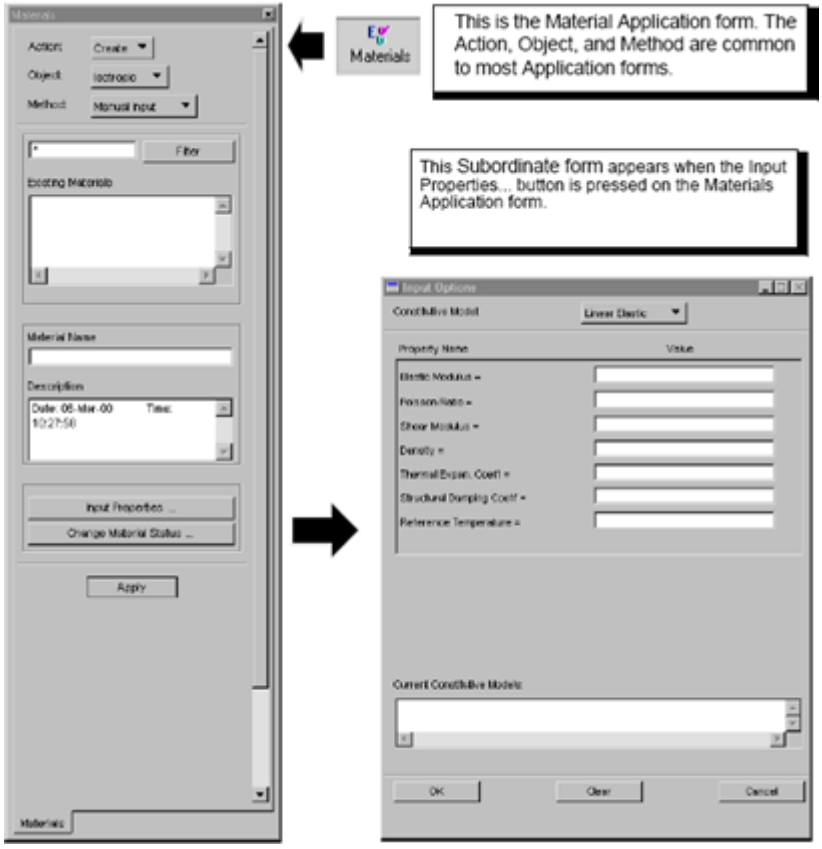
There are a number of application buttons in the Patran window that relate to specific tasks in Patran. For a standard FEA analysis, the radio buttons are arranged so that you can progress from left to right in the course of your MSC.Patarn session. However, you can access most buttons at any time, once you have created a database.

Brief descriptions follow for all the Patran application buttons.

Application	Function
Geometry	Creates and edits geometric models.
FEM (Elements)	Selects finite element shapes and creates a finite element mesh.
LoadsBCs (LBCs)	Assigns loads and boundary conditions.
Materials	Defines the material properties of the model.
Properties	Assigns element properties to model entities.
Load Cases	Defines load cases for your model.
Fields	Defines variation in properties and materials.
Analysis	Sets analysis parameters, submits the job, and reads result files.
Results	Defines how the results from the analysis display and plot.
XY Plot	Defines XY plots of results.

Application Forms

Selecting an application button displays an application form. Application forms are the primary method to define your model and control the analysis. Many application forms have subordinate forms, such as the Materials form and its subordinate Input Options form shown below.



You can choose only one application radio button at a time. Once you pick a second application from the window, the second application form appears and the first form closes. While you can only have one application form open at a time, you can have other forms open from the menubar. The tabs located at the bottom of the forms allow you to toggle between the forms that you have open. You can close an application by clicking its radio button.

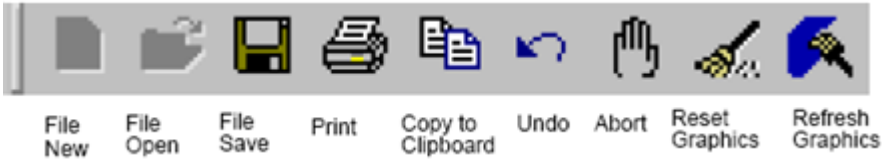
The Toolbar

The toolbar is a set of often-used functions displayed as a row of icons. These functions are grouped into several sets and used for model view control, render styles, predefined view orientations, and other functions. It is customizable, so you can add your own functions and icons or remove them.

You can rearrange the icons on the toolbar. Simply click and drag a set of icons to another area on the toolbar or to another location in the Patran window.

System Functions

The nine icons shown below are the system icons. At start up only two icons are active: the File New and File Open. Once a database is opened, the remaining icons become active.

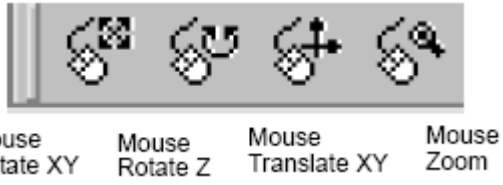


Icon Name	UNIX Icon	Function
File New	Use File/New	Brings up the New Database form.
File Open	Use File/Open	Brings up the Open Database form where you can select an existing database to open.
File Save	Use File/Save	Updates any changes made to your database.
Print	Use File/Print	Prints the contents of a viewport or the XY window. You can print a single viewport or window, or you may print multiple viewports or windows.
Copy to Clipboard	Not Available	Copies the contents of the current viewport to the clipboard.
Undo		-Apply- or -OK- buttons perform actions that add, delete, or modify entities in the database. To Undo the last operation invoked by -Apply- or -OK-, press the icon that depicts a reversal arrow. The graphics will refresh, all entities that were deleted as a result of the last apply are redisplayed, any entity that was added is erased, and any entity that was modified is returned to its previous state.

Icon Name	UNIX Icon	Function
Abort		There may be times during a session when you would like to abort a time consuming operation. This can be accomplished by depressing the icon depicting a raised hand. If the heartbeat is spinning when depressing the icon, Patran responds by presenting a form asking, “Do you want to abandon the operation in progress?” This question requires a yes or no response.
Reset Graphics		Removes all fringe and marker plots, all automatic titles, highlighting, and deformed shape plots. Repaints the viewport in wireframe mode. This button works on all posted viewports in entity mode, but only on the groups posted in the current viewport in group mode.
Refresh Graphics		Refreshes all of the graphics viewports.
Default Window Layering		UNIX only. Brings the Main form to the front of the display screen and layers other menus and viewports. This is especially useful when the Main form has obscured the viewport or other menus.
MSC Information		UNIX only. On Windows use Help/About Patran. This includes the Patran release number, your customer name and ID number, the X server and X Client information, trademarks, copyright information, and access to the license status program.

Mouse Functions

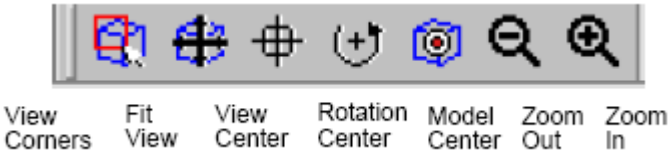
Using the toolbar icons below, you can customize the middle mouse button to change to view of your model.



Icon Name	UNIX Icon	Function
Mouse Rotate XY		Sets the middle mouse button to control model rotation about the X and Y axes.
Mouse Rotate Z		Sets middle mouse button to control model rotation about the Z axis.
Mouse Translate XY		Sets middle mouse button to control model translation in the X and Y directions.
Mouse Zoom		Sets middle mouse button to control zoom in and out of the model.

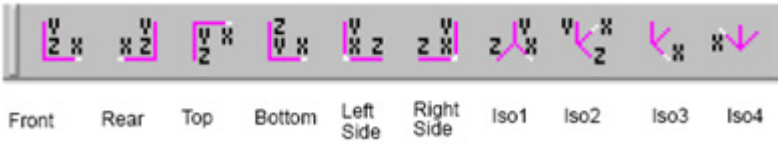
Viewing Functions

The Viewing menu located on the top menubar contains several options for controlling the view of your model in the viewport. A subset of these options are quickly accessible from two sets of viewing icons located on the toolbar.



Icon Name	UNIX Icon	Function
View Corners		Zooms in on a cursor-defined rectangular area in viewport.
Fit View		Resizes the current view so that all entities display in the current viewport.
View Center		Pans the model by moving the current viewport center to a cursor-picked location.
Rotation Center	Not Available	Allows selection of the model's rotation center by choosing a point, node, or screen position.
Model Center	Not Available	Sets the rotation center to the centroid of the displayed entities.
Zoom Out		Incrementally zooms out from the model by a factor of two.
Zoom In		Incrementally zooms in on the model by a factor of two.

The next set of viewing icons define the orientation of the model in the viewport. By clicking one of the icons below, you can quickly view your model from different angles. The default orientation is a front view.



Icon Name	UNIX Icon	Function
Front		X = 0, Y = 0, Z = 0
Rear		X = -180, Y = 0, Z = -180
Top		X = 90, Y = 0, Z = 0
Bottom		X = -90, Y = 0, Z = 0
Left Side		X = 0, Y = 90, Z = 0
Right Side		X = 0, Y = -90, Z = 0
Iso1		X = 23, Y = -34, Z = 0
Iso2		X = 23, Y = 56, Z = 0

Icon Name	UNIX Icon	Function
Iso3		X = -67, Y = 0, Z = -34
Iso4		X = -157, Y = 34, Z= 180

Display Functions

The two sets of Display icons located on the toolbar provide quick changes to the way in which you display your model in the viewport.

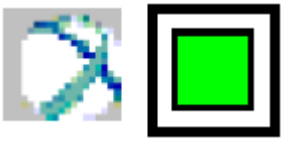
Icon Name	UNIX Icon	Function
Wire Frame		Displays model in wireframe render style.
Hidden Line		Displays model in hidden line render style.
Smooth Shaded		Displays model in smooth shaded render style.
Show Labels		Plots all entity labels.
Hide Labels		Erases all entity labels.

Icon Name	UNIX Icon	Function
Plot/Erase		Displays Plot/Erase form.
Label Control		Displays a form for controlling label display for selected entities.
Point Size		Toggles point display size between one and nine pixels.
Node Size		Toggles node display size between one and nine pixels.
Display Lines		Toggles the geometry visualization lines between zero and two.

Home (Windows only)

Click on the Home icon to bring up MSC Software's Web page using your default browser.

Heartbeat



The Heartbeat icon indicates whether Patran is busy or waiting for input from you. A stationary MSC globe on Windows machines means ready and waiting. A spinning MSC globe with a blue boarder means

busy, but can be interrupted. A spinning MSC globe with a red boarder means busy, cannot be interrupted. On UNIX it is a solid square with the same color meanings.

Command Line and History List Area

The Command Line and History List Area appear at the bottom of the Patran window. You can manually enter commands in the command line. The history list area is used to view the commands that Patran generates when a menu form is executed, and to view errors or informative messages. You can resize the History window in order to see more lines in the History List Area.

Picking and Selecting

Many times when you are working with application forms, a menu of icons appears to the left of your form. From this Select menu you can choose objects directly off the screen rather than typing in data on the application form. By selecting one of the Picking Filters from the Select menu, you control what type of objects you pick off the screen and how you select the object.

Picking and selecting off the screen can be a complex task. This section covers basic information about several aspects of the Select menu.

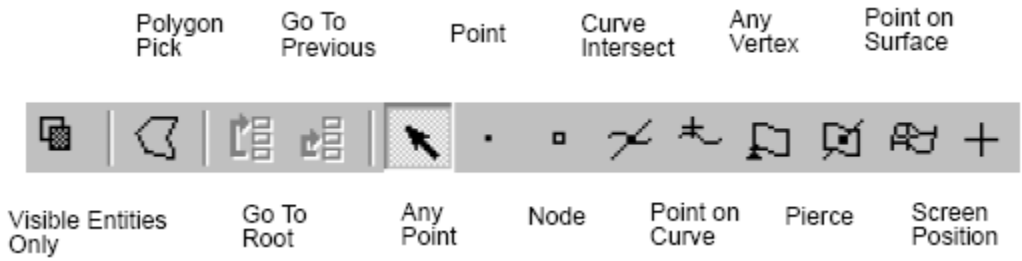
- Types of Select menus.
- Different levels within Select menus.
- Menu appearance.
- How to pick entities from the screen.
- Picking operations.

Select Menus

There are over 25 different Select menus in Patran. The Select menu that appears on your screen depends on the application in which you are working. For example, if you are creating a new point in your geometry model, the Point Select menu appears. Alternatively, if you want to select a finite element from a mesh that you have already generated, you will see the Element Select menu on your screen.

Each Select menu displays a series of graphic icons. As you move the cursor over each icon, the filter name appears next to the icon. The first four icons are common to all Select menus and control overall picking and navigating functions. The remaining icons are specific to each Select menu and act as picking filters

An example of the Point Select menu is shown below.



Icon	Function
Toggle Visible Entities Only Selection	Selects either visible objects only, or visible and hidden objects.
Polygon Pick	Selects all objects that lie within a polygon area.
Go to Root Menu	Reverts back to the original Select menu.
Go to Previous Menu	Reverts back to the previous Select menu.
Any Point	Selects a point associated with any geometric or finite element object.
Point	Selects a point.
Node	Selects a node.
Curve Intersect	Selects the point at which two curves intersect.
Point on Curve	Selects a point on a curve closest to a point off the curve.
Any Vertex	Selects the vertex of a curve, surface, or solid.
Pierce	Selects the point where a curve and a surface intersect.
Point on Surface	Selects a location on a surface.
Screen Position	Selects a X-Y screen position.

Multilevel Select Menus

Some of the Picking Filter icons you select will display a second and perhaps a third level Select menu. Each lower level menu in the hierarchy automatically replaces the previous Select menu. Only one Picking Filter menu is active at a time. As you continue to make filtering selections you further define how you will pick the object off the screen.

To navigate through different levels of menus in the hierarchy, use the two icons near the top of each Select menu:

Click on the Go to Previous Menu icon to return to the previous Select menu

or

Click on the Go to Root Menu icon to return to the original Select menu

Appearance of the Select Menu

By default, the Select menu appears as a set of icons arranged vertically and docked in the application form area. Like Patran's toolbar, you can undock and reposition the Select menu as a free floating menu at any time. You can also dock the Select menu on the left side of the window, in the toolbar area, or at the bottom of the window. Any changes that you make to the appearance or location of the Select menu remains in effect each time you open a database.

To change the appearance of the Select Menu:

1. Move the cursor over the title bar area of the Select menu.
2. Click and drag the menu to a new location.

   The Select menu can appear as a free floating menu anywhere in the viewport.

or (on Windows)

1. To maintain a vertical set of icons, click and drag the Select menu to the left side of the Patran window and release.
2. To switch to a horizontal set of icons, click and drag the Select menu to the toolbar area or to the History/Command Line area.

Picking an Entity off the Screen

There are two basic methods of picking entities off the screen: enclosing an entity by drawing a boundary around it, or pointing directly at the object, and selecting it.

Enclosing Entities

You can use the enclose function to select any number of objects off the screen at one time. This is especially useful when you want to select a large number of objects that you would otherwise have to select one by one.

On the Select menu there are two types of boundaries that you can use to enclose objects: a rectangular boundary and a polygon boundary. The rectangular boundary is the default. The polygon boundary offers more flexibility in defining the shape of the boundary.

To use the Polygon boundary:

1. Click on the Polygon icon located on the Select menu (on UNIX this icon is in the main toolbar).
2. First move the cursor into the application form databox and click.
3. Next move the cursor into the viewport. Click once to define the first vertice of the polygon.

   From there you can move and click the cursor as many times as is necessary to define your polygon.

To use the rectangular boundary:

1. Click in the application form databox and then move the cursor into the viewport.
2. Click and drag the cursor arrow to define an enclosing rectangle.

What happens when an object lies partially within and partially outside the boundary you drew? This depends on your picking preferences. Using the Preferences/Picking menu from the menu bar, you can specify that the object needs to be entirely enclosed, partially enclosed, or only the centroid need be enclosed. This preference then instructs Patran whether to include or exclude an object lying on the boundary.

Selecting Individual Entities

On each Select menu there are typically several icons that control how you pick individual objects off the screen. These features are most useful when you are selecting one or two objects, or when the desired objects are located such that you can't enclose them using a boundary.

The filtering icons are specific to the application. The default picking icon is the Any icon. Using this function you can select any object off the screen that corresponds to the current application. This is essentially the unfiltered picking function.

To select an object:

1. Click in the application form databox and then move the cursor into the viewport.
2. Point the cursor arrow at the desired object and click.

or, to be more selective in your picking:

1. Select one of the picking filter icons on the Select menu.
2. Click in the application form databox and then move the cursor into the viewport. Point the cursor at the desired object and click.

Changing the Picking Operation

Using the Select menu you can add an entity to an application form, replace an existing entity on the form, or remove an already selected entity form a form. When the Select menu first appears, it is in Replace

mode. Any entity that you select from the screen replaces an existing entity if one is designated in the databox.

To change this mode to add or remove entities:

1. From the Patran menu bar, select Preferences/Picking.

   The Picking Preferences menu appears in the Application form area.

2. Click on the Show Picking Icons checkbox.

   This instructs Patran to add a set of Picking Operation icons to your Select menu. You will see a message box informing you that you need to restart Patran in order for this change to take place.

3. Quit and restart Patran.

4. Resume the Application that brings up your Select menu.

You should now see the three additional picking operation icons. These icons remain on each Select menu until you change the Picking Preferences form again.

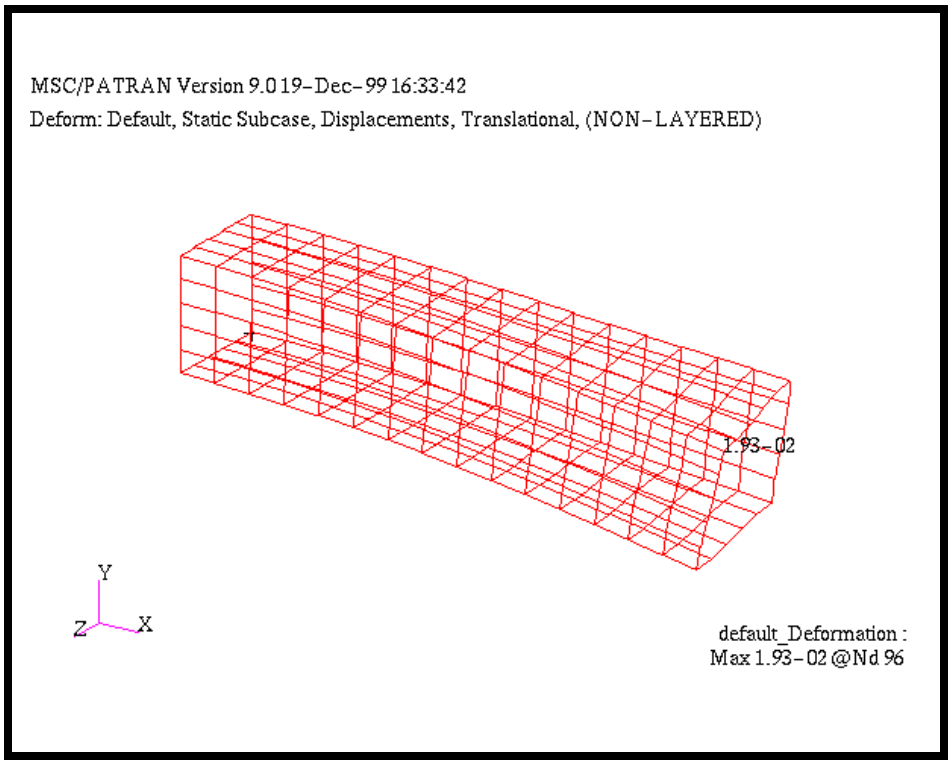
5. Click on one of the other Picking Operation icons to change the Picking mode.

This operation changes a setting in a file called settings.pcl, generally located in the installation directory. If this operaiton does not work it is usually because there are no permissions set to change this file. You may need to either change the permissions or copy the file to the directory from which you are running and do the operation again.

Picking objects from the viewport can be a difficult task. The features discussed in this section are the basic features of picking and selecting. For more information about individual picking icons, select menus, and picking off the screen, please refer to the *Patran Reference Manual, Volume 1, Part I* Introduction to Patran, Chapter 3: Interactive Screen Picking.

Exercise 1

Introduction to MSC.Patran



Objectives:

- Create geometry for a Beam.
- Add Loads and Boundary Conditions.
- Review analysis results.

Notes about this exercise:

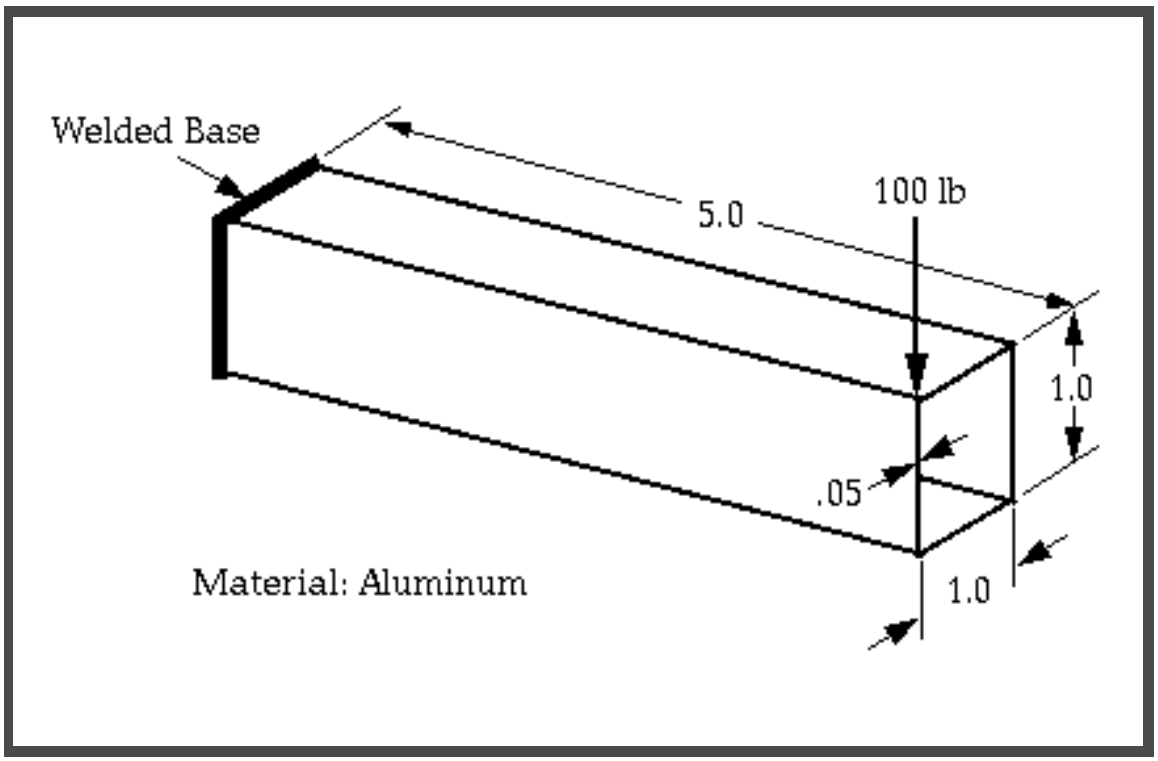
- This document contains specific instructions for a slightly different version of Patran than we have available on the EWS machines. Discrepancies between this document and our Patran interface will be indicated with comments in dashed boxes.

- This document gives instructions to perform analysis with MSC sftware, however we will use Abaqus for our analysis. Sections 16-18, which pertain to analysis and results, will be replaced with other Abaqus instructions.

Model Description:

This exercise will take you through the steps of modeling an open-ended aluminum box beam that is welded to a rigid surface. It has dimensions that are shown in the diagram below.

A downward 100-pound force will be applied, as shown. You will determine how much the box beam will deflect, as well as the maximum von Mises stress.



Suggested Exercise Steps:

- Create a new database and name it as **box_beam.db**.
- Create geometry.
- Apply boundary conditions and loads.
- Analyze model and view results.

Exercise Procedure:

*Note: In most MSC.Patran forms, the default setting for the **Auto Execute** button is on; thus, you do not need to press **Apply.***

Open a New Database

1. Create a new database and name it **box_beam.db**.

File/New...

New Database Name **box_beam**

OK

Navigate to the directory you want in the New Database Dialogue Box.

You can save to your home directory sub directories in your home directory on the EWS machines.

New Model Preferences

Tolerance: **Default**

Analysis Code: **MSC/PATRAN_FEA**

Analysis Type: **Structural**

OK

Analysis Code: ABAQUS

No Mapping

Note: If the analysis code MSC/PATRAN_FEA is not available to you, choose another structural analysis code for which you have licenses.

If the part of the screen which was covered by the New Model Preferences form is not redrawn, press the refresh button:

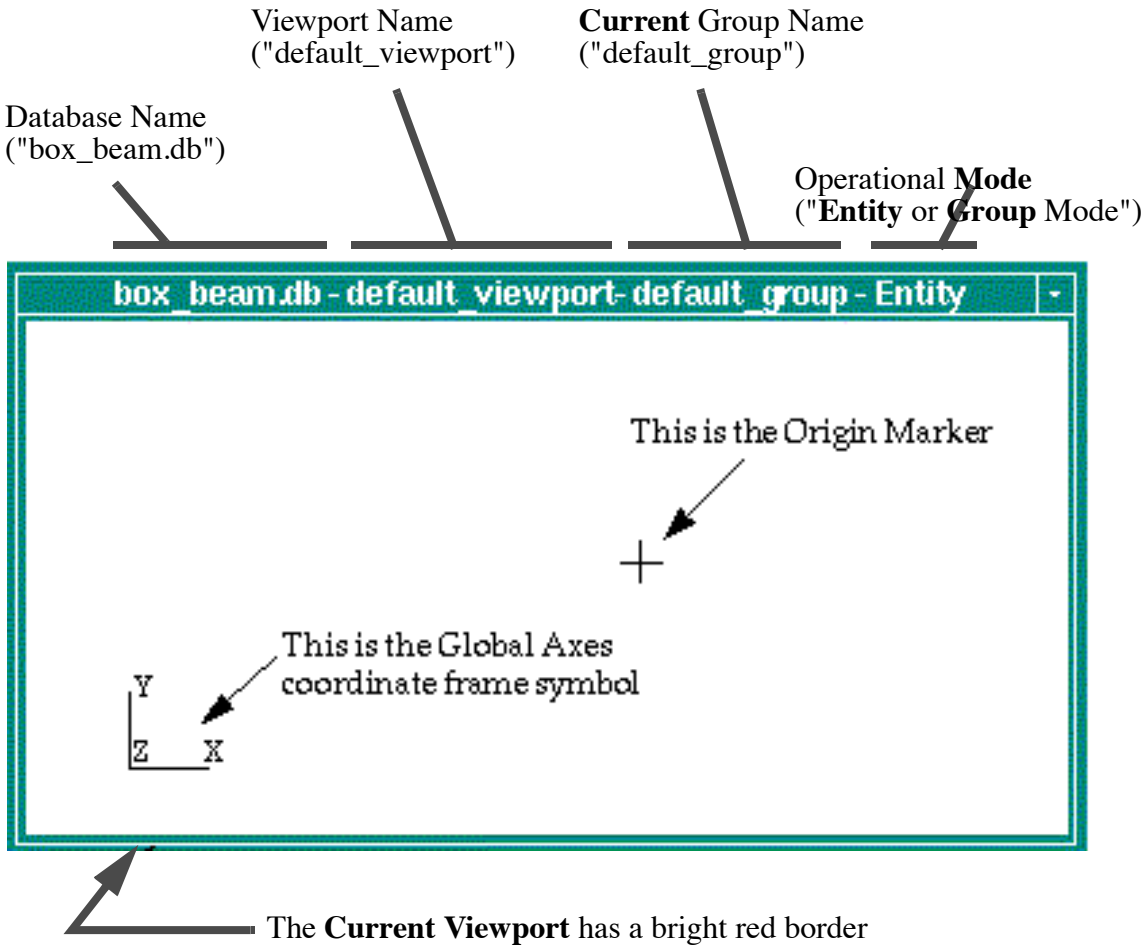
To change the viewpoint color:

- Viewport
- Modify
- Change "Background color" in "Attributes" box

2. A Short Word on Viewports and Groups

After pressing **OK** on the *New Model Preferences* form, you should see a large graphics window, called the MSC.Patran **Viewport** (see below).

A viewport is a window in which you view your model. You can display more than one viewport, but to keep it simple we will be working with only one.



The top of the viewport lists the name of database you are working on, the name of the viewport, the name of the current group you are working with, and the mode of operation for displaying the model (Entity or Group mode).

A **Group** is a defined collection of geometry and/or finite element entities. Entities, such as a surface or an element, can belong to more than one group.

The **Current Group** is a specified group to which all newly created entities will belong. You can only specify one group at a time to be the current group for each viewport.

Create a Solid

3. Create a Solid

Though the box beam we will be creating is hollow, we will create a solid of the box beam and later, mesh the outside surfaces of the solid.

Keep in mind the dimensions of the beam are 5 inches long with a 1 inch cross section. We will create the solid with the long direction in the +X direction.

◆ **Geometry**

Action:	**Create**
Object:	**Solid**
Method:	**XYZ**
Vector Coordinates List	**<5, 1, 1>**
Origin Coordinates List	**[0, 0, 0]**

Apply

Try rotating the model about the XY axis using the middle mouse button

Experiment with view type, e.g. wire frame, hidden line, etc.

For fun, rotate the model by placing the cursor in the viewport and holding down the middle mouse button and dragging the mouse around. Notice that when you release the button a View/Fit View is automatically performed. You can switch from the rotate function to translate or zoom by entering Preferences/Mouse...

Change the Viewing Angle

4. Change the View Angle.

Instead of using the middle mouse button to change the view of the model, you can use the Viewing menu, by doing the following:

Viewing/Angles...

◆ **Model Absolute**

Angles	**20 -10 0**

Apply

Cancel

Change the Viewing Angle

5. Reset the Number of Display Lines

All surfaces and faces of solids are shown with additional lines, called **Display Lines**. These are not actual geometric boundaries, but they are **visualization** lines to help you see the interior curvature of the surfaces and faces.

Sometimes you may need to simplify the display of your model by lowering the number of Display Lines, and this is done as follows:

Display/Geometry

Number of Display Lines **1**

Apply

Cancel

6. Create the Displacement Boundary Conditions.

Create Boundary Conditions

MSC.Patran has this wonderful ability to associate the loads and boundary conditions (Loads/BCs) either with the geometry or with the finite element model.

Very important!

The Loads/BCs associated with the geometry will be applied automatically to the nodes and elements when they are created. If you decide later to remesh the model, the Loads/BCs will be automatically revised. This means you can remesh your model as many times as you wish and you won't have to worry about reassigning the Loads/BCs to the mesh - MSC.Patran will take care of it for you!

For our box beam model, we want to "fix" the welded end of the beam by defining no movement in all six degrees of freedom, which is represented by six zeroes. (A null or blank value, instead of a zero, means the specific degree of freedom is free to move.)

To apply the fixed boundary condition:

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name **fixed**

Input Data...

Translations <T1,T2,T3> **< 0 0 0 >**

Rotations <R1 R2 R3> **< 0 0 0 >**

Analysis Coordinate Frame **Coord 0**

OK

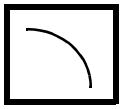
Select Application Region...

Geometry

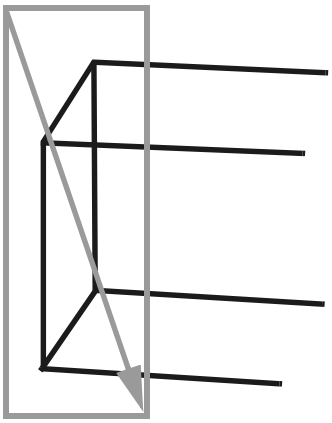
Select Geometry Entities

At the left of the menu you should see a select menu. By default the Geometric Entity icon is highlighted, which means MSC.Patran will look for all geometry entities when you cursor select entities in the viewport.

But we want to cursor select only the edges of the solid for our fixed boundary conditions. Thus, click on the Curve or Edge icon:



Cursor select the left end edges of our solid by using a rectangular cursor selection:



With the cursor at point "A", **hold down** the left mouse button and **drag** the cursor to point "B", so that you have drawn a rectangular box around the left end edges of the solid, as shown. Then, release the mouse button.

Note: MSC.Patran highlights the selected edges in red, and the "Select Geometry Entities" databox will list the solid edge IDs. (i.e., Solid 1.1.1, 1.1.2, 1.1.3, etc.).

If you are not happy with what you have selected, you can deselect the edges by using the **right** mouse button.

You may need to change your picking option here:

- Preferences
- Picking
- Select "Enclose entire entry" in the "Rectangle/Polygon Picking" box

Alternatively you can shift+click to select multiple edges

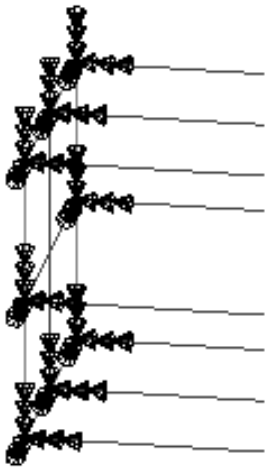
Finish creating the boundary condition by pressing the following menu buttons:

Add

OK

Apply

You should see light blue “cones” being drawn on the end of the solid at the Display Lines’ point locations (shown below). These represent the three fixed translational and the three fixed rotational boundary conditions. Remember, these have been applied to the geometry!



If you ever need to come back and view the boundary conditions again go to:

- Load/BCs
- Action: Plot Markers
- Select loads or BCs you want to view in the "Assigned Loads/ BCs Sets" box
- Select "default_group" in "Group Filter" box
- Click apply

7. Apply the 100-pound Force Load.

Apply Load

Now we want to finsh our Loads/BCs by applying our 100-pound downward force on the right corner of the solid. The force will be defined by a vector in the global rectangular coordinate frame, where we will have **zero pounds in the +X direction**, **100 pounds in the -Y direction** and **zero pounds in the +Z direction**.

Again, we don’t need to create the finite element mesh before applying our force. We can associated the force with the solid.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name	**100_pounds_down**
Input Data...	
Force <F1 F2 F3>	**< 0 -100 0 >**
Analysis Coordinate Frame	**Coord 0**

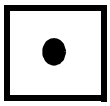
OK

Select Application Region...

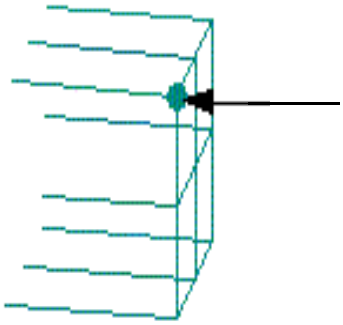
Geometry

Select Geometry Entities

Since we want to apply our force to a point on the solid, switch the select menu icon to the **Point or Vertex icon**:



Cursor select the corner point of the right end of the solid, as shown:



Click on this corner of the solid with the left mouse button. MSC.Patran will highlight the point in red.

You should see “Point 7” entered in the “Select Geometry Entities” databox. If you cursor selected another point by mistake, press the right mouse button to deselect it.

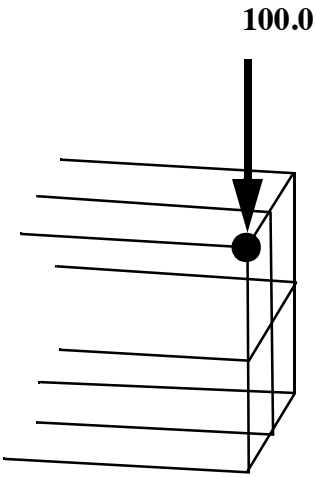
Finish creating the force by pressing the following menu buttons:

Add

OK

Apply

Now you should see a yellow arrow drawn on the selected corner point of the solid which is our 100-pound force:



8. Create a New Group

Create a New Group

Before we go on and mesh our box beam model, we want to keep our soon-to-be finite element mesh in a group that is separate from our geometry model.

To do this, we will created a new, empty group called “fem_model” and make “fem_model” our current group.

Group/Create...

New Group Name **fem_model**

Make Current

Group Contents:

Add Entity Selection

Entity Selection

Apply

Cancel

Note: “fem_model” has replaced “default_group” at the top of our viewport, which means “fem_model” is our new current group.

Define Mesh Seeds

9. Define Mesh Seeds on one end of the Solid.

Before we mesh the outer surfaces of our solids, we want to take advantage of a feature called **Mesh Seeds.**

Mesh Seeds allow you to define exactly how many elements (and even what node spacing) you want on selected curves or edges of a surface or a solid.

For our box beam model, we want to define mesh seeds of five elements in the Y and Z directions and 15 elements in the X direction. To do this, we do the following:.

Just called "Elements"

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

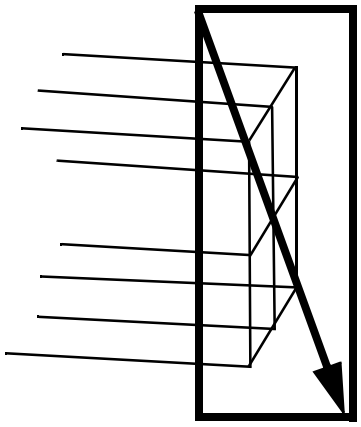
◆ **Number of Elements**

Number=	**5**
Curve List	

In our version you can select either

Because mesh seeds can only be associated with curves or with edges of surfaces or solids, the select menu only has one icon. This means MSC.Patran will only look for any curves or edges that you select.

Since we want to select the edges of the solid that are in the Y and Z directions, use the rectangular cursor select box and cursor select the edges on the **left end** or the **right end** of the solid (the illustration below shows the solid's right end edges being selected):



Use the rectangular cursor selection box and select the edges on the left or right end of the solid

If auto execute is selected, do not click apply.

If you accidentally click apply, select NO when the warning box asks if you want to duplicate seeds.

Apply

Now we want to create our second mesh seed for 15 elements in the X direction:

◆ **Number of Elements**

Number=	**15**
Curve List	

Shift + click to select all four edges

Select one of the 4 edges in the *middle* area of the solid. The selected edge of the solid will be **red**.

Apply

10. Mesh the Outside Faces of the Solid with Quad4s

Meshing

Now we're ready to create our finite element mesh. Though we have a solid, we want to create a surface mesh made of 4-noded quadrilateral elements (Quad4s). MSC.Patran allows you to create a surface mesh on the *faces* of our solid.

To do this, do the following:

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	**0.1**
Element Topology:	**Quad4**

◆ **IsoMesh**

Surface List	

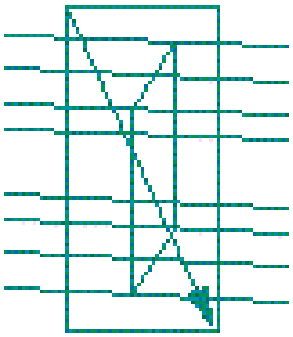
The order of these objects is different on our version of Patran.

Select "Quad" for "Element Shape" and "Isomesher" for "Mesher"

Notice the select menu will only allow you to select surfaces or solid faces, because the Type option on the Finite Elements form is set to "**Surface".** Again, either individually select the outside solid faces (not including the two ends) using shift/click, or:

you may now cursor select the faces in the *middle area* of the solid, using the rectangular cursor selection box:

We need to change the picking preference to "Enclose part of entity" to select the middle area of the solid. See note on page 1-8

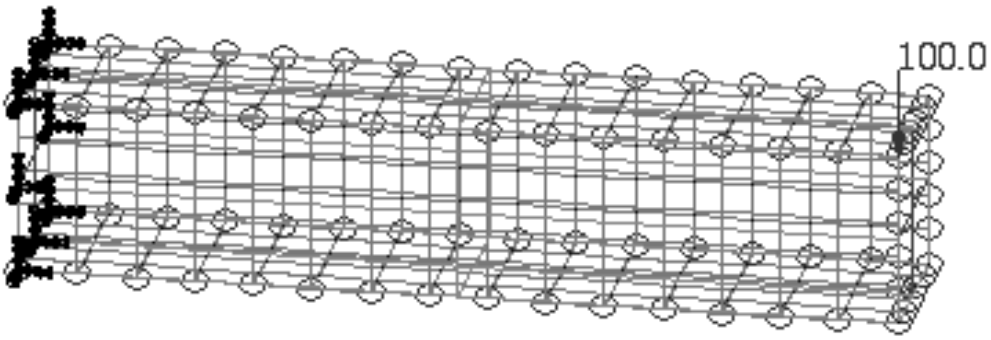


Use the rectangular cursor selection box and select the edges in the **middle** of the solid. Use Preferences/Picking and select Enclose any portion of entity. The selected edges of the solid will be red.

The Surface List databox should list "solid 1.3 1.4 1.5 1.6".

Apply

The model should now look like the following:



Equivalence

11. Equivalence the Coincident Nodes.

You may not realize it, but duplicate or coincident nodes were created along the edges between the neighboring solid faces. You will need to equivalence the nodes to remove each on of the two coincident nodes.

◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Type:	**Tolerance Cube**

Equivalencing Tolerance **0.005**

Apply

Purple circles will appear which tell you where the coincident nodes were found and removed.

Element Normals

12. Align the Element Normals.

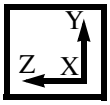
"Why bother?", you may ask? You may not realize it, but 2-D shell elements, like the Quad4s used in this box beam model, have a defined top and bottom surface for results output.

By default, the Quads4s we created have their positive surface normals (which are the elements' top surfaces) pointing in the positive global coordinate directions. But for postprocessing the results, we want the normals to be pointing outward from the box, regardless of their orientation in global XYZ space.

To show you what we mean, let's first plot the existing normal vectors by doing the following.

First, reset the view by looking down on the end of the box beam

Click on the Right Side View icon:



Now let's plot the element normal vectors:

◆ **Finite Elements**

Action: **Verify**

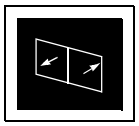
Object: **Element**

Method: **Normals**

Display Control

◆ **Draw Normal Vectors**

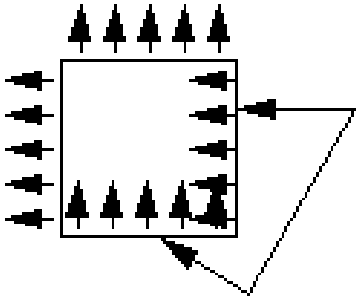
Make sure the Test Control icon looks like this:



It should state "Display Only" on the side

Apply

You should see red arrows being plotted that look like this:

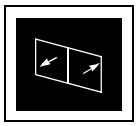


We want to reverse the element normal directions on these two sides of the box so they point outward.

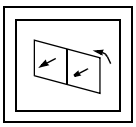
Notice the normals are aligned with the +Y and +Z global coordinate directions.

Now, let's reverse the normal directions of the two sides indicated above.

First, press the **Display Only** icon, under **Test Control**, so that it changes to the **Reverse Elements** icon:



Changes to:

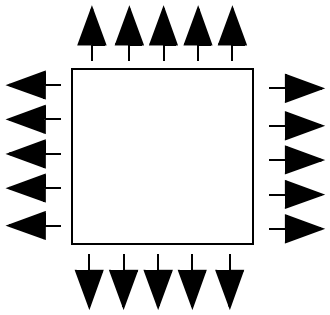


Guiding Element

Use the left mouse button and pick one element from the top surface which is pointing in the "correct" or outward direction.

Apply

You should see the element normals all pointing outward:



The normal vectors' red arrows can be erased by either pressing the Reset Graphics button, or by exiting the Finite Elements Application and continuing on to the next step.

Define Material Properties

13. Define the Material Properties.

Since the box beam is made of aluminum, we can assume the material is isotropic and it will behave in the linear elastic region. To create the material property:

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**aluminum**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus=	**10.0e6**
Poisson Ratio=	**0.3**
Density=	**0.000259**
Thermal Expand. Coeff.=	

Just "Elastic"

Click "OK" to dismiss the "Input Properties" box

Apply

Cancel

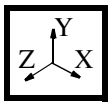
Define Element Properties

14. Define the Element Properties

Now we will define the element properties for our Quad4 elements. The elements have a thickness of 0.05 inches and should reference the aluminum material property that we have just created.

We will associate the element properties with the geometry model, and not the finite elements. This will allow MSC.Patran to reassociate the properties to the elements if we later decide to remesh the model.

First let's reset the view to make it easier to cursor select our elements. Press Iso 1 View icon from the toolbar:



Now let's create our element property and reference it to our geometry model:

Properties

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**box_beam_shell**
Option(s):	**Homogenous**
	Standard Formulation *

* May not be an option depending on your analysis preference.

Input Properties...

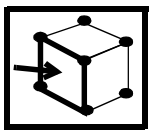
Material Name	**m:aluminum**
Thickness	**0.05**

OK

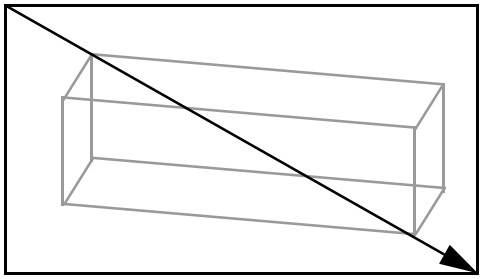
Click "Select Application region…"

Select Members

Since we want to associate our element properties with the outside surfaces (faces) of our solid, make sure the Surface or Face icon is highlighted in the select menu:



Now cursor select the entire model (remember our solid is still in the group, "default_group", which is still posted (displayed) to the viewport) Click on the Iso 1 View icon at the top to see the whole model:



Select the entire model using the rectangular selection box.

You should see "Solid 1.1 1.2 1.3 1.4 1.5 1.6' appear in the databox which are the solid face ID's.

Add

Apply

Click "OK" to exit "Input Properties" box

15. Create a Load Case

Create a Load Case

Now we want to create a load case which groups selected loads and boundary conditions into a single set which can be referenced for the analysis.

◆ **Load Cases**

Action: **Create**

Load Case Name **(load case description)** load_case_1

Make Current

Load Case Type: **Static**

Description **load_case_1**

Description is optional

- Click "Input Data…"
- Select "Disp_fixed" and "Force_100-pound_down" to add to spreadsheet

Now click on the Assign/Prioritize Loads/BCs button.

Select Loads/BCs to add to spreadsheet **Disp_fixed**

Force_100-pound_down

OK

Apply

Analyze / Job Status / Results

We will skip steps 16-18 in this document, follow modified steps at end of this document instead

16. Perform the Analysis / Check Status of Job / Read Results

Now we are ready to submit our finite element model for analysis.

Under Analysis, we will set up the job for the analysis and submit it directly from MSC.Patran to MSC.Nastran.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Job Name **box_beam_loadcase_1**

Translation Parameters...

Solution Type...

Direct Text Input...

Subcase Create...

If you have time, you may want to look at the subordinate forms for these options. The defaults for these options are fine for our model.

Subcase Select... **load_case_1**

OK

Apply

After pressing Apply, the heartbeat on the main form will turn blue and you should see a number of messages scroll by in the Command Line. The heartbeat will then turn to green which means you can continue executing operations in MSC.Patran while the analysis is running.

To see if MSC.Nastran has completed with no errors, open another window and search the contents of the file, *box_beam_loadcase_1.f04* in the directory where you started up MSC.Patran for the string EXIT:

For UNIX, enter:

% more box_beam_loadcase1.f04 | grep EXIT

For Windows NT, enter:

% type box_beam_loadcase1.f04 | find "EXIT"

If MSC.Nastran completed with no errors, you should see the following line:

16:23:52 0:20 20.2 0.0 11.4 0.0 SESTATIC 145 EXIT BEGN

Once the analysis has successfully completed, it will produce a *box_beam_loadcase_1.op2 results file. This file must be read into the MSC.Patran database before you can begin to postprocess the results.*

Analysis

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**
Available Jobs	**box_beam_loadcase_1**
Select Results File...	
Available Files	**box_beam_loadcase_1.res**
OK	
Apply	

You should see messages appear in the Command Line, stating which results are being read into the database.

Now we are ready to postprocess the results.

17. Unpost (Erase) the Geometry Group.

Unpost Geometry Group

Since we are done using the geometry, let's erase or "unpost" the group, "default_group", which contains the geometry part of our model:

Group/Post...

Select Groups to Post **fem_model**

Apply

Cancel

Notice the Loads/BCs symbols go away because they are associated with the geometry that is part of "default_group", which is now unposted.

Create a Deformation Plot

18. Create a Deformed Shape Plot

Let's create a deformed shape plot based on the displacement results. This is an excellent way to view the response of our structure. Note: The numbering of the result cases may vary.

Results

Action: **Create**

Object: **Deformation**

Select Result Case(s)

Select Result **1.1-DISPLACEMENTS, TRANSLATIONAL**

Apply

Although there are other ways to reset the graphics display of our model back to the default wireframe, the easiest way is to press **Reset Graphics** icon at the top *before* creating another postprocessing results plot.

Create a Fringe Plot

Start here after completing analysis with Abaqus and creating the deformation plot

19. Create a Fringe Plot of the von Mises Stresses.

Next, let's create a color fringe plot of our von Mises stresses. Invariant stresses, such as von Mises stresses, are a good choice for a fringe plot since they are scalar quantities, and they are a good predictor of the yield stress in metals like aluminum.

◆ **Results**

Action:	**Create**
Object:	**Fringe**
Select Result Case(s)	
Select Fringe Result	**Stress...**
Quantity:	**von Mises**

"Stress, components"

Apply

20. Create a Combined Fringe/Deformed Plot

Let's create a contour plot of our membrane stresses on top of a deformed shape plot by doing the following:

You can view/hide the undeformed configuration by clicking the "Display Attributes" icon near the top of the "Results" pane, then select/ unselect "Show Undeformed" and click "Apply"

◆ **Results**

Action:	**Create**
Object:	**Deformation**

Select "Deformation, Displacements" in the "Select Deformation Result" box

Apply

Now you should see the contour plot on top of the deformed shape plot.

21. Modify the Finite Element Mode (Optional)

Step 21 will not be covered in the lab sessions

Note the high stress regions are bending stresses in the opposing corners at the box beam's open end (opposite of the welded end). But they are well below aluminum's 30 ksi yield stress.

Being the good engineer that you are, you realize the design could be greatly improved by welding a cap on the open end of the beam.

Try posting the geometry model ("default_group") and mesh the solid face on the open end. You can then quickly equivalence and optimize the model and assign the existing element properties to this new area.

Run the model through MSC.Nastran again and see if the stress levels improve. Good luck!

Close the Database and Quit MSC.Patran

22. Close the Database and Exit MSC.Patran.

File/Quit

Changes to your model will be automatically saved when you close your database or exit MSC.Patran.

Congratulations and give yourself a big pat on the back!

You have just had a glimpse of the future -- the way productive people will be doing finite element analyses from now on.

And there's much more! Try accessing a CAD model, creating varying loads and element or material properties through the use of Fields, interpolating results from one analysis onto another model as loads, and using the Finite Element Sweep mesh creation.

Examples of these and other features of MSC.Patran can be found in Part 10: Example Problems in the MSC.Patran User's Manual.

PERFORM THE STEPS BELOW INSTEAD OF PATRAN STEPS 16-18 IN THE BOX_BEAM EXERCISE

Create the *Abaqus* Input File for Box Beam Model

All aspects of the finite element model are now defined. Thus the *Abaqus* input file can be generated.

1. On the main menu across the top of the screen (on the row of diamond buttons), select the one for ♦**Analysis** (push the diamond button using the mouse and left mouse button).
2. This brings up a new **Analysis** menu form on the right side of the screen and dismisses the **Loads/BCs** menu form. In the new form, use the left mouse button to pick the following entries:
   Action: Analyze
   Object: Entire Model
   Method: Analysis Deck
3. Click the **Job Name** box, then type **box_beam** (the job name is user definable. The Abaqus input file will then be named box_beam.inp).
4. Click the **Optional Controls** button. This brings up the **Optional Controls** menu.
5. Change the Results File Type: to FIL and ODB. Click **Apply**.
6. Click the **Step Creation** button. This brings up the **Step Create** menu.
7. Select the **Default Static Step** in the **Available Job Steps** box.
8. Click the **Output Requests** button. This brings up the **Output Requests** menu which lists various kinds of stress, strain, displacement, etc. output that can be requested.
9. Change the Form Type: from Basic to Advanced.
10. For BOTH Stress **Component (S)** and **Strain Component (E)**, we want to change the Element Position: from **Integration Pts** to **Nodes**. This will request that *Abaqus* output values at the nodes of each element. Click **Ok** on the **Output Requests** menu to save requests and dismiss the menu.
11. Click **Apply** (click **Yes** if asked whether to overwrite), then the **Cancel** button in the **Step Create** menu. This saves values and dismisses the menu.
12. Click the **Apply** button on the remaining main **Analysis** menu. This causes *Patran* to actually write the *Abaqus* input file. For larger models this can require as much as 10 seconds. During this process, the *Patran* graphics window disappears. When the heartbeat (at the top right corner of the screen) turns green and the graphics window reappears, the task is finished. The *Abaqus* input file is now named *box_beam.inp* (based on the responses given above).
13. If you would like to take a look at how the input file was generated, open the input file named *box_beam.inp* with a text editor. In the Terminal for example, type

```
gedit box_beam.inp
```

Other options are:

```
gvim box_beam.inp
nano box_beam.inp
```

Hint: If you are using `nano`, you can press `Control+W` and perform a search for the keyword you are looking for. This comes in handy in large files. Emacs has the same functionality too.

Run *Abaqus* to Perform Analysis

14. In an X-terminal window, change to your work directory then initiate execution of *Abaqus* using the command:
    `abaqus -j box_beam`
    Abaqus now starts execution to perform the analysis.

    ***NOTE1**: If this throws a "-bash" error, type: `module load abaqus`
    This will load abaqus to your directory.

    ***NOTE2**: A second option for running Abaqus is to simply type `abaqus`
    Abaqus will run interactively and ask for the `*.inp` filename
    `identifier : box_beam`

    The program runs in the 'background'. This enables continued use of the X-terminal window. *Abaqus* is a fast program for linear analysis and most jobs for our classes complete quickly. It will take around 10-15 sec to finish it, although in X-terminal window it would seem like it was executed in a second. To check the status type *ps* in the X-term. If you see *box_beam.com* under the list of the ps commands that would mean, the program is still running.

    Several key files produced by *Abaqus* during execution (using *box_beam* as the identifier if given above, otherwise they will have the same filename):
    - *box_beam.dat* contains the printed error messages, results and other comments
    - *box_beam.log* contains a very short summary (20 lines) of the starting and stopping steps in the analysis. Simply display this file to determine when your analysis is completed. Use the command: `cat box_beam.log`
    - *box_beam.fil* contains the results in binary form for *PATRAN* to read.

Import Results Into *Patran* Database

In this step, we will 'import' the nodal displacements, stress and strain into the *Patran* database for this model and generate a graphical image of the deformed shape. Perform the following sequence of actions:

15. On the main menu across the top of the screen (on the row of diamond buttons), select the one for **♦Analysis** (push the diamond button using the mouse and left mouse button).
16. This brings up a new **Analysis** menu form on the right side of the screen. In the new form, use the left mouse button to pick the following entries:
    Action: Read Results
    Object: Result Entities
    Method: Translate
17. Click the **Select Result File** button.
18. Select on the proper result file in **Available Files** window (*box_beam.fil* in this example).
19. Click **OK** on the **Select File** menu.
20. Click the **Apply** button on the **Analysis** menu. This causes *Patran* to actually read the *Abaqus* results file into the *Patran* database (box_beam.db) for this problem. For larger models this can require as much as 10-20 seconds. During this process, the *Patran* graphics window disappears. When the heartbeat turns green and the graphics window reappears, the task is finished.

Generate a Deformation Plot

This step produces a graphics image of the deformed shape superposed over the undeformed shape. Then we can generate a hardcopy output of the image.

21. On the main menu across the top of the screen (on the row of diamond buttons), select the one for **♦Results** (push the diamond button using the mouse and left mouse button).
22. This brings up the **Results** menu form on the right side of the screen and dismisses the previous menu.
23. Examine the **Select Result Cases** 'box'. There should be available selections. Select the bottom one in the 'box'. Examine the **Select Deformation Result** 'box'. Select the option **Deformation Displacements**. Click the **Apply** button at the bottom of this menu.
24. The undeformed image changes to a have blue lines, with the deformed image in white color lines for triangle elements.
25. Now we will 'animate' the result as thought the deformed shape is a 'mode' shape from a vibration analysis. Click the **Animate** button just above the **Apply** button on the **Results** menu. Click **Apply** on the **Results** menu. The **Animate** menu pops up over the **Results** menu and the graphics image cycles dynamically through the deformed shapes. Click the **Pause Animation** when desired.

Clean-up Image and Generate Fringe Plot of Deformed shape

The graphics image is cluttered with the undeformed plot, deformed plot plus all the points, curves and surfaces from the original geometry. The following sequences clean up the image in several stages.

26. On the main menu across the top of the screen, select **Group→Modify**. This brings up the **Group** menu form on the right side of display. Find the **All Geometry** line with has two buttons below the label. Click on the **Remove** button, then click the **OK** button.
27. Click the 'broom' (**Reset graphics**) icon on the top-right side of the main menu. Then return to step 21 to again plot the deformed shape.
28. Select the "Object: **Quick plot**" and **Default** for the **Select Result Case**. Click on **Deformation Displacements** under Select **Fringe Results** and under **Select Deformation Result**. Click on **Apply**.
    Even though the geometry items are now invisible, the undeformed mesh remains. This can be suppressed with the following sequence.
29. Below the "Object: **Quick plot**" scroll menu click on the **Deform Attribute** icon (as hover over these buttons the name of the button appears). This changes the menu form below. Find the **Show Undeformed** button. Click it 'off' (i.e. the button is out).
30. Click **Apply** on the **Results** form to apply this change and to dismiss the form.
31. Click the 'broom' icon on the top-right side of the main menu. Then return to step 21 to again plot the deformed shape. Now only the deformed shape is shown when requested. The 'broom' icon can be used to tidy-up the graphics image at any time.

BACK TO PATRAN DOCUMENT STEP 19

OPTIONAL: Hardcopy of Graphics Image

The current graphics image can be routed to the PostScript printers at any time during a *Patran* session. This includes the geometry model, the finite element model (with constraint and load markers), the deformed shape, etc. The sequence below may be followed at any time to send an image to the printer (or to generate a PostScript file for e-mail, ftp, sending to a remote printer, to examine in Ghostview, etc.)

1. On the main menu across the top of the screen, select **File→Print**. This brings up the **Print** menu form on the right side of display.
2. Find the **Page Setup** button on the **Print** menu. Select this button to bring up a menu form which controls the page layout. Examine the options and select the ones you want (selecting none of these is ok). Click **OK** to dismiss this menu.
3. Find the **Options** button on the **Print** menu. Select this button to bring up a menu form which controls some additional options. In this menu, you need to select "color" in

Format, "white" in **Background** and "actual" in **Lines and Texts**. Besides, select **Print to File** option.

4. If you would rather have an EPS file, click on **♦Create EPS File**. For example, Adobe Illustrator can directly open EPS files. If not, a PostScript file will be created. GSview for example can open both PS and EPS files.
5. Click **Ok**. Click on the **Apply** button of the **Print** menu. This causes the hardcopy process to execute. Once the heartbeat turns green again, the plot has been sent to a file.

***NOTE:** There are a few useful command line commands. Probably the most useful one is

`epstopdf file.eps`

You could also explore the `ps2pdf`

Some quality and layout differences may occur when using these converters.

UNIVERSITY OF ILLINOIS AT URBANA-CHAMPAIGN

College of Engineering

CEE570/CSE551 — Finite Element Methods (in Solid and Structural Mechanics)

Spring Semester 2014

Common Unix C–Shell (or TC–Shell) Commands

`ls [options] directory`

List directory content. This command lists files and directories that reside in directory. If directory is omitted the current directory is used. Options: -l: long format, - a: include the dot files (filenames starting with.), -F: identify directories with a trailing /, executable files with a *: list subdirectories recursively, -1: force output to be one entry per line.

`cp source target`

Copy files. This command copies the source file into the target file. Options: -R: recursively copy directories.

`mkdir directory`

Make a directory. This command creates an empty directory with the name directory.

`mv source target`

Move a file or directory. This command renames or moves the source file to the target location. If the source is a directory name then only a rename operation is possible.

`rm [options] file`

Remove a file. This command removes the file. Options: -f: forcefully remove, do not attempt to verify, -i: interactive, prompt to remove each file, -r: recursive, delete files and subdirectories recursively.

`rmdir directory`

Remove a directory. This command removes the directory. The directory must be empty before this command will work.

cd *directory*

Change directory. This command changes the current directory to directory. Accessing the folder ".." (without quotes) is equivalent to going back one folder level.

pwd

Display name of working directory. This command displays the full path of the current directory.

man *unix-command*

Manual. This command gives information on unix-command.

Options: -k <keyword>:list all commands with this keyword in their one-line description.

ps [options]

Process status. This command lists the user's processes running on system. Options: -e: give process status on all processes in system.

chmod [operation][permission] *directory/filename*

Change access permission. Allows the user to change the read/write/execute permissions for user/ group/others.

cat *file*

Prints the contents of a file directly on the Terminal. Useful for viewing small text files. Do not attempt this with a large file.